Sborník vědeckých prací Vysoké školy báňské - Technické univerzity Ostrava
číslo 3, rok 2009, ročník LII, řada hutnická,
článek č. 1530

İsmail GERDEMELİ[1], Serpil KURT[2], Derya ÖZER[3]

MATERIAL STRENGTH ANALYSIS OF THE PARTS OF THE CAGE STRUCTURAL GIRDER GANTRY CRANE

MATERIÁLOVÁ PEVNOSTNÍ ANALÝZA ČÁSTÍ OCELOVÉ KOSTRY NOSNÍKU PORTÁLOVÉHO JEŘÁBU

[1] *Istanbul Technical University; Istanbul, Turkey, gerdemeli@itu.edu.tr*
[2] *Istanbul Technical University; Istanbul, Turkey, ozerde@itu.edu.tr*
[3] *Istanbul Technical University; Istanbul, Turkey, kurtserp@itu.edu.tr*

Abstract

Nowadays, nearly in all industrial fields cranes are used. Especially, in heavy industries, such as, in iron steel production, in mold industry, in shipyards etc. The transportation of heavy constructions are provided with the gantry cranes to the assembly line. In order to prevent the damage of the cranes during lifting heavy loads, strength calculations of the cranes are investigated and compared with the results obtained from finite element analysis. In this study, after a short description of the gantry crane parts, the technical specification of the cage–main girder gantry crane are explained. The crane parts are modelled and afterwards, the strength analysis of the crane parts are accomplished with Abaqus/CAE software and DIN standards. As a conclusion, the results obtained from finite element method and analytical calculation are compared.

1. Introduction

Cranes are widely used to transport heavy loads and hazardous materials in shipyards, factories, nuclear installations, and high-building construction and play an important role in production process and serve to transfer loads from one place to another. Gantry cranes are the best way of providing a heavy lifting facility covering virtually the whole area of the industry. Their design features vary widely according to their major operational specifications such as the type of motion, dead weights and type of the load, location of the crane, geometric features and environmental conditions. Since the crane design procedure is highly standardized with critical components, main effort and time spent mostly for interpretation and implementation of available design standards.

A gantry crane is a type of crane with a hoist in a trolley which runs horizontally along gantry rails, usually fitted underneath a beam spanning between uprights which themselves have wheels so that the whole crane can move at right angles to the direction of the gantry rails. These cranes come in all sizes, and some can carry and move very heavy loads, particularly the extremely large examples are used in shipyards or industrial installations.

In this study, a gantry crane with a cage beam is modelled in 3D using Autocad and Solidworks computer software.Then, the cross section of gantry crane parts are checked by accomplishing the strength equations due to FEM, DIN norms.

Considerable research studies have been carried out about structural and equivalent stresses in order to provide safety under static loading and dynamic behavior of cranes. Finite element analysis is a powerful technique originally developed for numerical solution of complex problems in structural mechanics, and it remains the method of choice for complex systems.[1] The basic principal of this numerical method is dividing all the large structure into small elements having simple shapes. The unknown variables of an element are the displacement values for each nodal point.

In this study, finite element model of the gantry crane parts with 550 ton capacity is completed using four node quadratic shell elements with Hypermesh Software. The stress and displacement analysis of crane have been achieved by considering each part of the gantry crane individually in ABAQUS 6.7.1. The system behavior is utilized for stress analyses.

2. Main Body

The specifications of gantry crane are the max. height of hoist as 45.000 mm and the hoisting capacity is as 2 x 275.000daN. The carriage walk span is 52 m and the walk span is 495 m. The forces affecting to the crane have been illustrated in the free body diyagram as in figure 1 and the strength calculations of the crane parts have been accomplished. The dead loads of the gantry crane, work load, dynamic loads and wind load are considered in these calculations that are achieved due to the DIN and FEM norms. Consequently, the cross -sections of the parts of the crane have been checked.

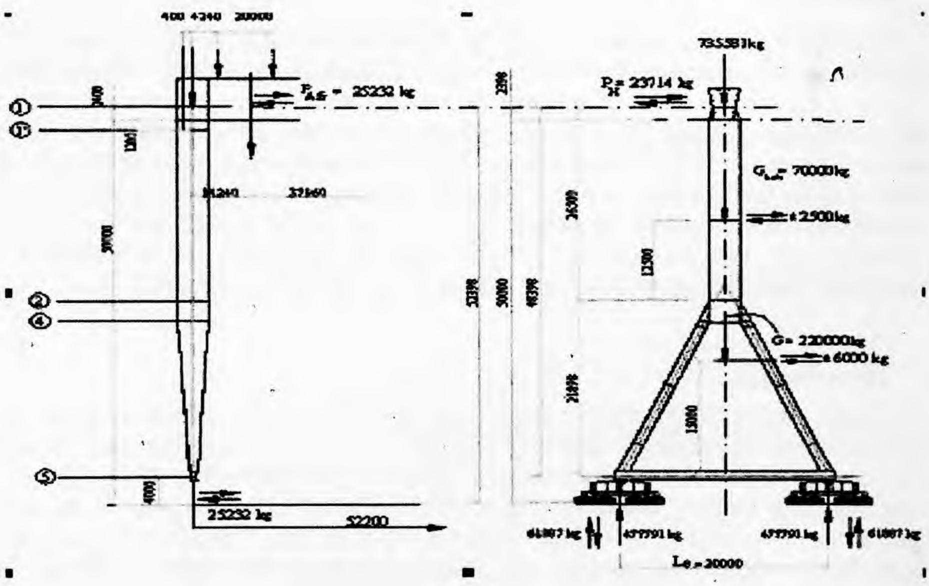

Fig. 1 The free body diagram of the gantry crane

For the finite element analysis, the element type used in the model is four node quadratic shell element (S4R). The total node number used is 392745 and the element number is 398602 for the gantry crane which is considered. All of the parts are meshed seperately and then, they are combined to each other with node to node connection. The gantry crane parts are composed of St42 material and the material properties of St42 are described in table 1.

Table 1 Material properties of gantry crane parts [2]

MATERIAL PROPERTIES	
Young's modulus of elasticity	***2.1x10⁵ N/mm²***
Poisson's ratio	***0,3***
Density	***8 ton/m³***

During the finite element analysis, some assumptions have been done. The nodes on the contact points of the boggie with the rail are constrainted. 2 nodes are considered as restrained and 32 nodes are free. Each carriage has 8 boggies and load is applied to the nodes on the contact points of the boggie with the ground. The total load is divided equally on 16 boggie and the carriage is assumed as static. Considering these assumptions and the material properties in table 1, the gantry crane parts are meshed as seen in figure 2.

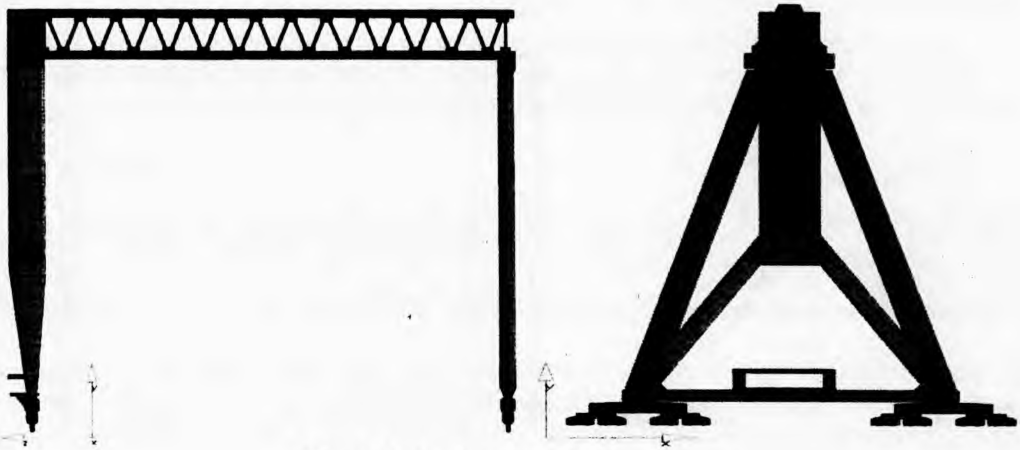

Fig. 2 Meshed model of the gantry crane

3. Results

Static analysis is the most common analysis method which is used in engineering. As the loads are assumed to be applied instantly, the effects due to the time variation are neglected.

The comparison of the results which are achieved by finite element method and lineer calculations are illustrated in table 2. The deviation between these two methods are not more than 20%. This deviation occurs because of the assumptions in analytical methods and the numerical approach used in the finite element method. As the computed stress values are smaller than the allowable stress of the used material (180 MPa), it is observed that the gantry crane is safe according to DIN and FEM norms.

Table 2 The comparison of the analytical and FEM method results

Control Regions	**Analitical Analysis (N/mm^2)**	**Analysis with FEM (N/mm^2)**
Tower upper side	99	97-109
Tower lower side	103	97-108
Equalizer Beam	168	155-177
Equalizer Beam	163	155-177
End Carriage	144	125-136
End Carriage	166	125-136

4. Conclusions

In this study, the stress values on winkle supports which are employed for safety against wrinkle proves that they do not carry load and they do not provide strength for tension, compression and bending. Because of this, the strength of these parts can be decreased. For instance, the tower wrinkle supports, upper and middle sheet metal of the assembly box, middle sheet metal of the upper box of the leg, boggie upper sheet metal and upper sheet metal of bottom cross beam, the support sheet metals placed in the main beam are not under effect of high stress values. The sheet metal thickness of these parts mentioned can be decreased.

Considering the datas that are obtained from the analyses, the material waste can be prevented in crane design. The construction is now more reliable, light and durable. This is crucially important in means of low cost production and low design duration.

References

[1]. David V. Hutton, Fundamentals of Finite Element Analysis, 1st Editon, McGraw-Hill Mechanical Engineering, 2004,

[2]. http://www.e-pipe.co.kr/eng/DIN/17177.htm

Reviewer: Prof. Ing. Miroslav Tvrdý, DrSc., VŠB –TU Ostrava